

# Conceptual Modeling Tutorial



2013.2

# Visual MODFLOW Flex

*Integrated Conceptual & Numerical Groundwater Modeling*

- Import a standard USGS MODFLOW data set (MODFLOW-2000, -2005)

[Start the Tutorial!](#) 

### **See Also:**

Several sample projects are available for download from our website, that illustrate both the conceptual and numerical modeling workflows

## **3.1 Conceptual Modeling**

The following example is a quick walk through of the basics of building a conceptual model and converting this to a numerical model.

### **Objectives**

- Learn how to create a project and import your raw data
- Become familiar with navigating the GUI and steps for conceptual modeling
- Learn how to define a 3D geological model and flow properties
- Define boundary conditions using your GIS data
- Define a MODFLOW grid, then populate this grid with data from the conceptual model
- View the resulting properties and boundary conditions
- Translate the model inputs into MODFLOW packages and run the MODFLOW engines
- Understand the results by interpreting heads and drawdown in several views
- Check the quality of the model by comparing observed heads to calculated heads

### **Required Files**

---

- Several files are required for this exercise, which should be included with the VMOD Flex installation. These files are available in your "My Documents" folder, VMODFlex\Tutorials\ConceptualModel. If you cannot find these files, please download them from our website:

<http://trials.swstechnology.com/software/VMODFlex/2012/Tutorials/conceptual-model-supp-files.zip>

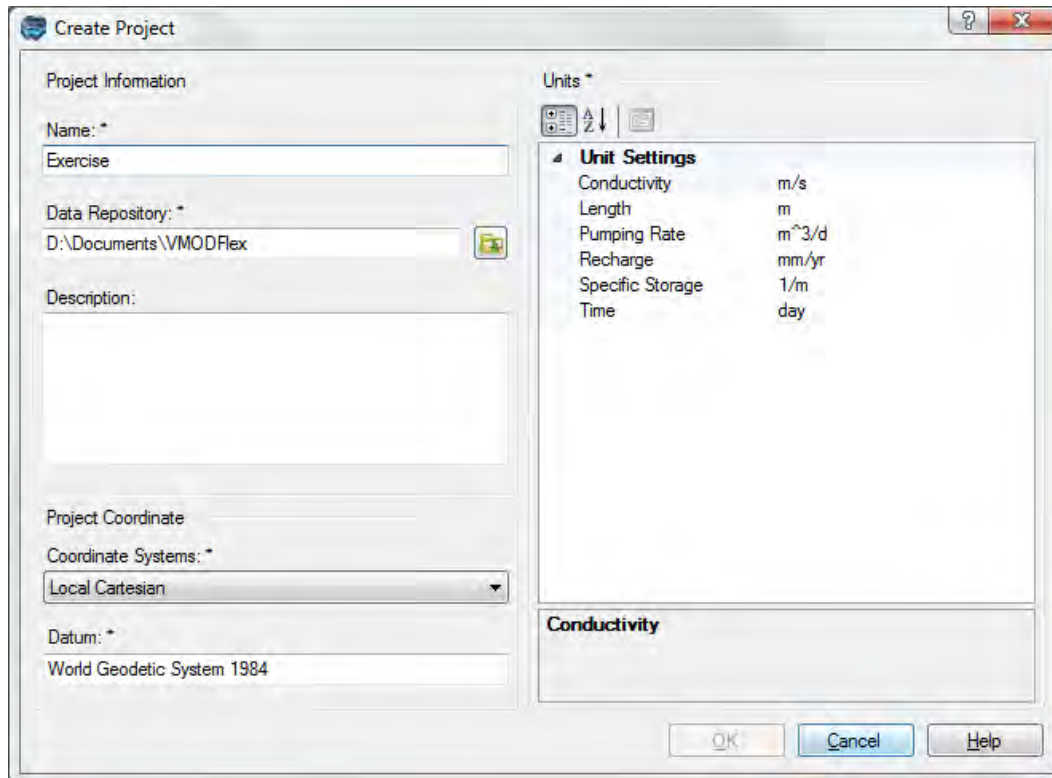
### **Creating the Project**

---

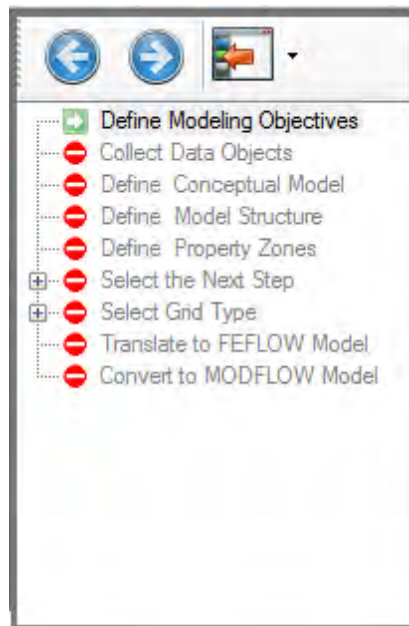
- Launch VMOD Flex
- Select File? New? Project. The Create Project dialog will appear.
- Type in project Name "Exercise".

- Click Browse button, and navigate to a folder where you wish your projects to be saved, and click OK
- Define your coordinate system and datum (or leave the non-cartesian as defaults)
- Define the Units frame. For this project, the default units will be fine.

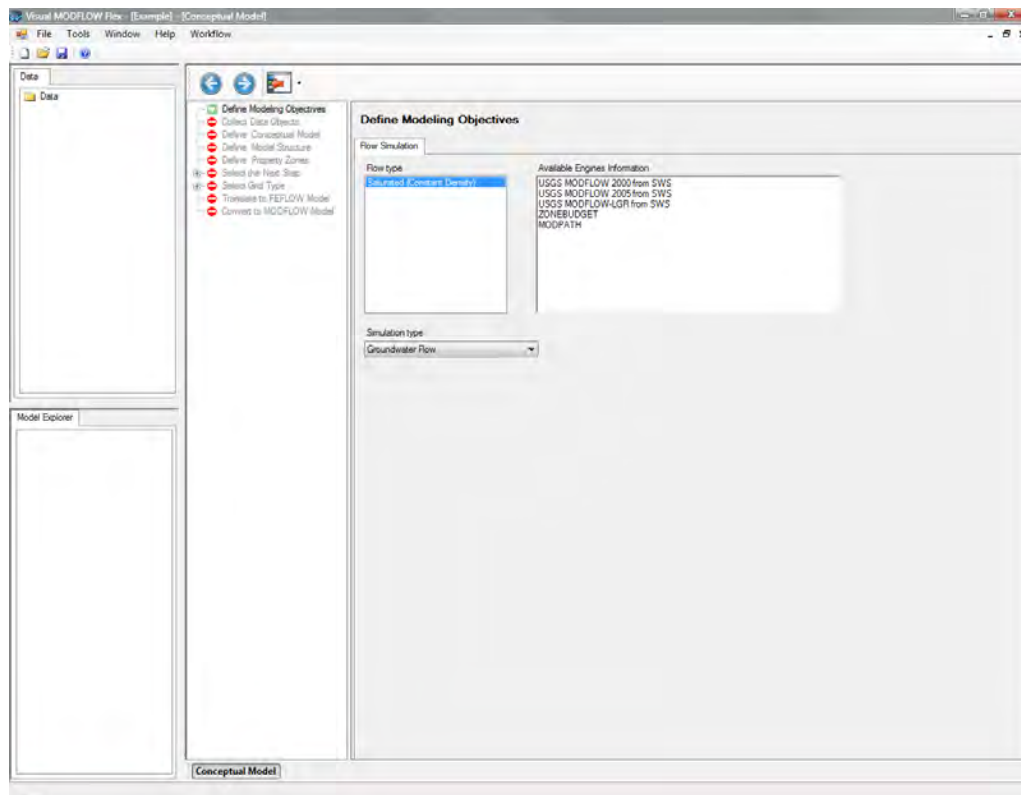
The Create Project dialog should now look like this:



- Click [OK]. The workflow selection screen will appear.
- Select Conceptual Modeling and the Conceptual Modeling workflow will load.



- In this step, you define the objectives of your model and the default parameters.

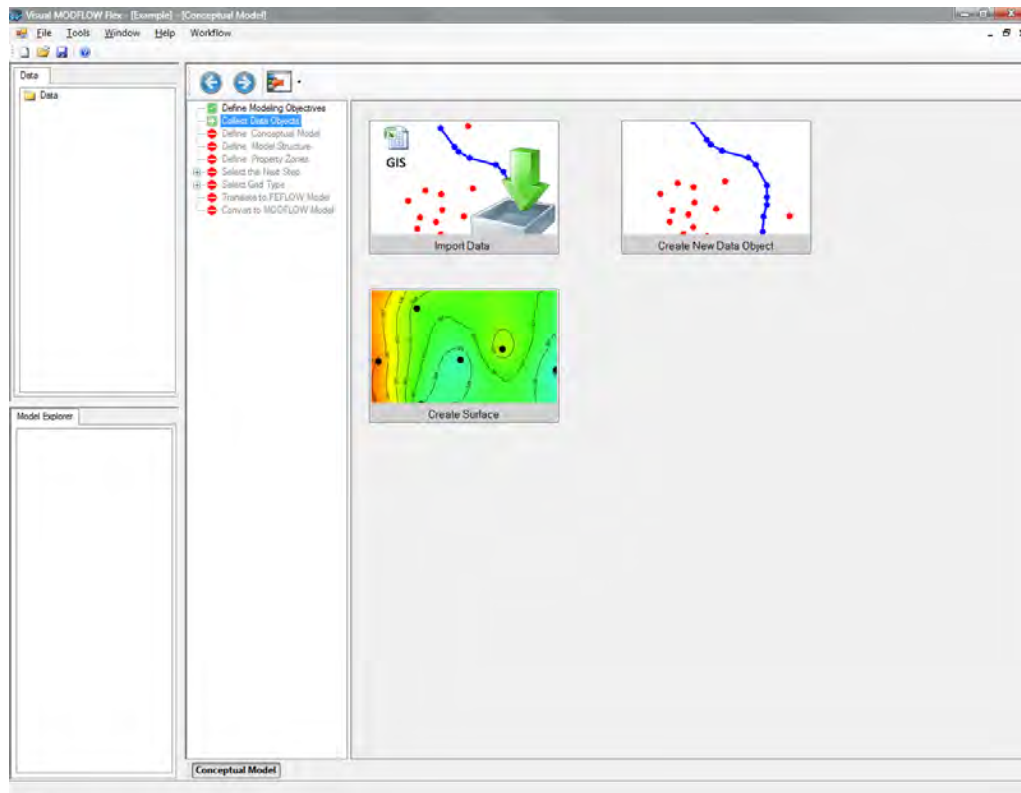


- For this scenario, the default objectives will be fine

- Click  (Next Step) to proceed.

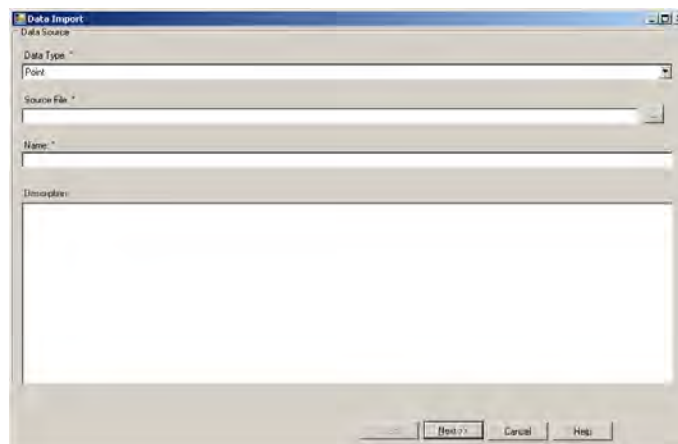
### Collect Data Objects

- The next step is to import or create the data objects you wish to use for building the conceptual model.

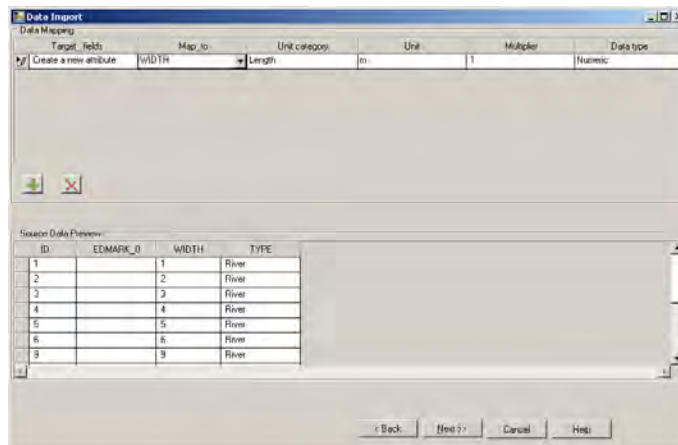


At this step, you can import data, create new data objects (by digitizing) or create surfaces (from points data objects)

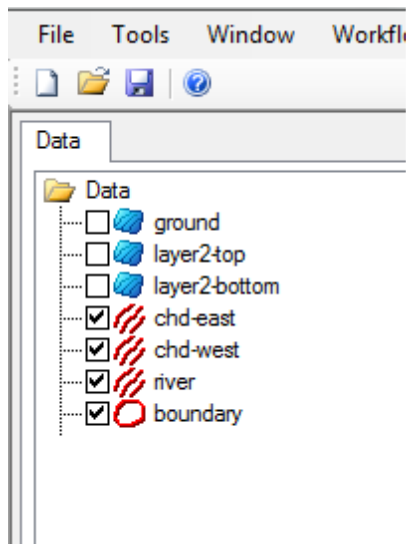
- Click the “Import Data” button and the following screen will load:




- Select Polygon in Data Type combo box.
- In the Source File field click the [...] button and navigate to your "My Documents" folder, then VMODFlex\Tutorials\ConceptualModel, select folder Polygons and select boundary.shp
- Click [Next>>]




- Click [Next>>] then Click [Finish]
- The next step is to import a surface that represents ground surface.
- Click the "Import Data" button
- Select "Surface" for the Data type.
- In the Source File field click the [...] button and navigate to the "My Documents" folder, then VMODFlex\Tutorials\ConceptualModel folder, and select ground.grd
- Click [Next>>] through all the screens to accept the defaults, then click Finish
- Repeat these steps to import the remaining Surfaces: layer2-top.grd, layer2-bottom.grd
- Next, import polyline data objects, and from the same source directory ,select chd-east.shp; use all the defaults and finish the import.
- Repeat these steps, for polylines, importing first chd-west.shp, then river.shp.
- Once the data objects are imported, they will appear in the tree on the left side of the program window.




- You can view these data objects in 2D or 3D; simply create a new viewer
- Click on Window/New 3D Window from the main menu; an empty 3D Viewer will appear;
- Click on the check box beside each of the data objects you imported, and they will appear in the 3D Viewer
- Click  (Next Step) to proceed, where you will arrive at the Define Conceptual Model step

### Define Conceptual Model

Provide a name for the conceptual model, start date, and model area.

From the Data Explorer, select the polygon data object that represents the conceptual model horizontal boundary, and then click the  button.



**Note:** The model area cannot be defined using a complex polygon, or one that contains multiple polygons. A complex polygon is a polygon that intersects with itself.

- Click  (Next Step) to proceed to the Define Model Structure step.

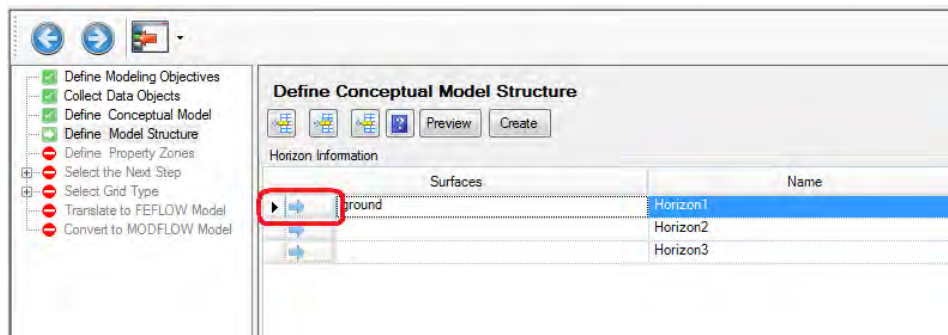
## Define Structure



Defining the geological model consists of providing geological surfaces as inputs for horizons; then, three-dimensional solids are created between these horizons.

To create new horizons, follow the steps below.

- From the Horizons Settings dialog (shown below), click the  Add Horizon button to add a new horizon row to the Horizon Information table.
- Repeat this two more times so there are 3 new rows on the Horizons table.
- From the Data Explorer, select the **ground** surface data object that will be used to generate the horizon
- Click the  button in **Row1** of the Horizons grid, to insert it into the Horizon Information table. See the example below.

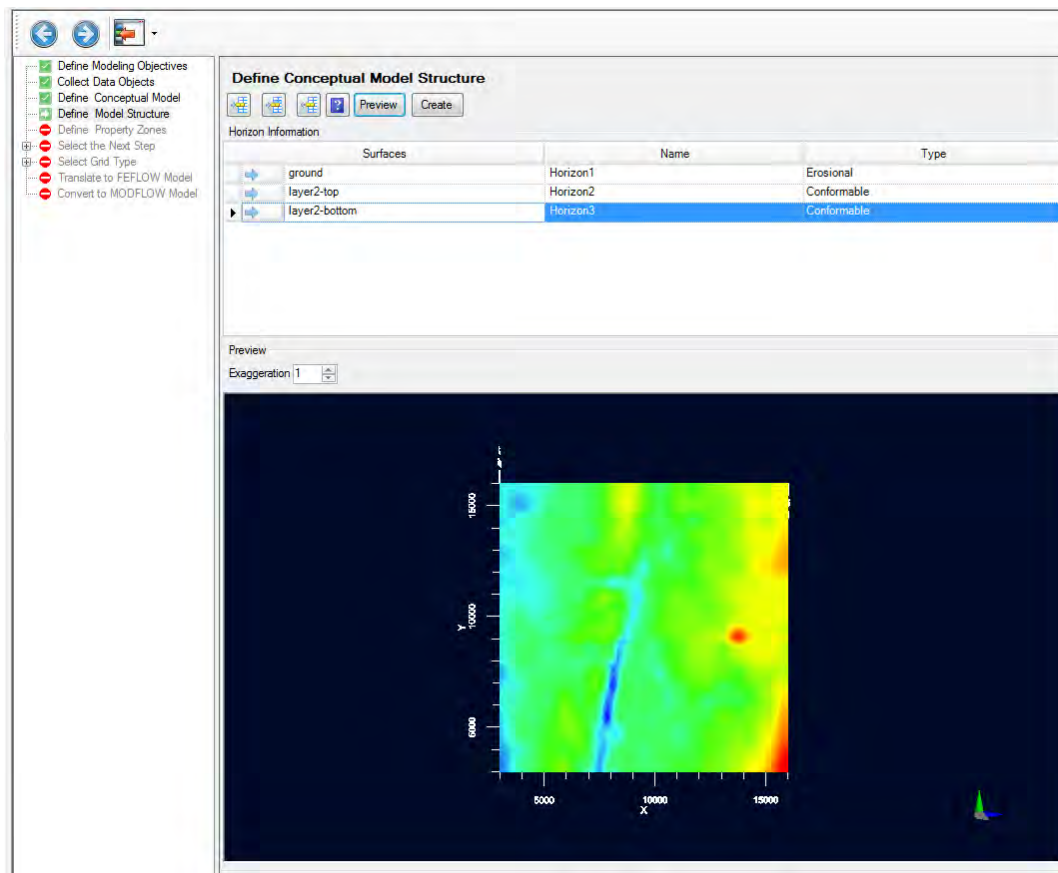





- For this example, the default horizon type will be adequate. For information on each horizon type, please refer to "[Horizon Types](#)".
- Repeat the steps above to add additional horizons:
- From the Data Explorer, select the **layer2-top** surface data object, click the  button in **Row2** of the Horizons grid, to insert it into the grid.
- From the Data Explorer, select the **layer2-bottom** surface data object, click the  button in the **Row3** of Horizons grid, to insert it into the grid.

**NOTE:** Horizons must be added from the topmost geological layers and working downwards.

- You can preview the horizons in the adjacent 3D Viewer, by clicking the [Preview] button.
- Once finished, you should see a display similar to the one shown below.



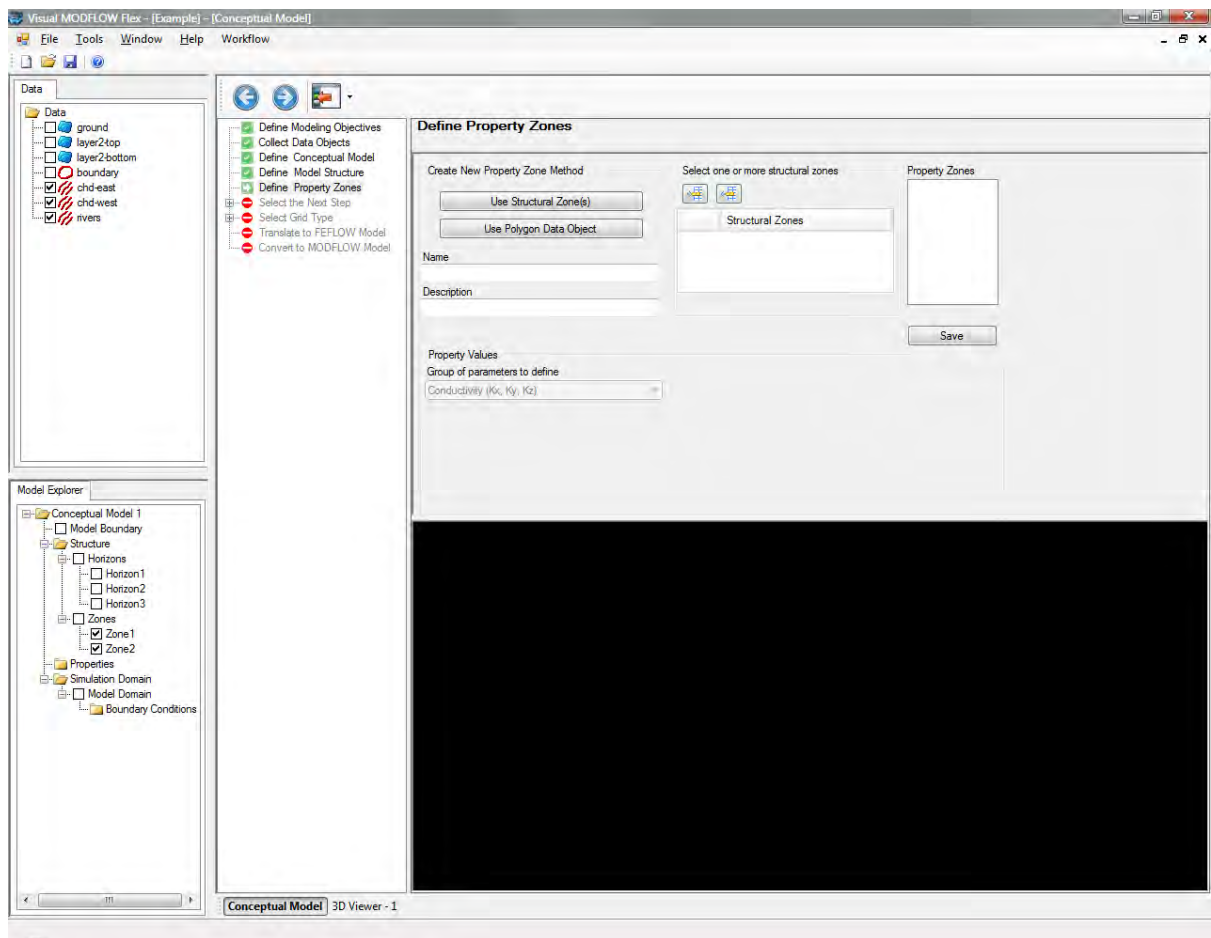
- Finally, click the [  ] button to create the horizons and proceed to the next step.

### Define Property Zones

Once you have imported sufficient raw data into your project, you can begin to construct one or more conceptual models using imported or digitized data objects as building blocks.

At this step, you can view/edit the flow properties for the model.

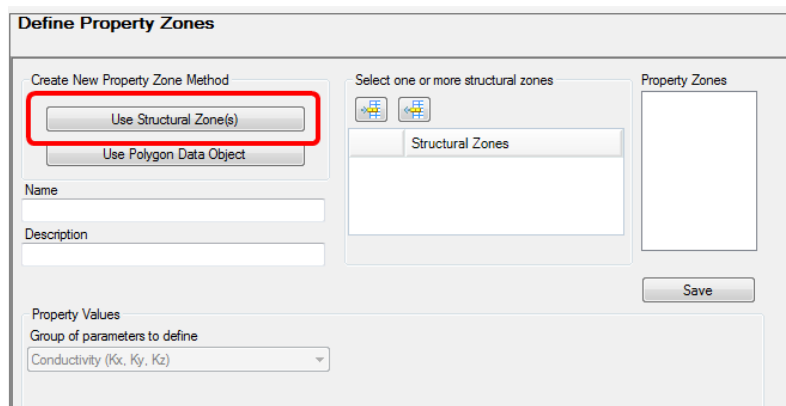
There are two ways to define property zones: Using Structural Zones, or Using Polygon Data Objects



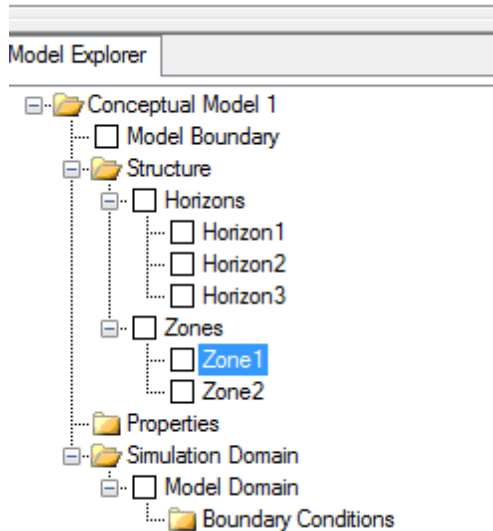
## Using Structural Zone(s)


This method allows you to create a property zone from existing structural zones in your conceptual model, i.e., zones generated from horizons.

- Click on the [Use Structural Zone] button as shown below



- Select Zone1 structural zone from the conceptual model tree (under the Structure/Zones node as shown below)



- Click the  button to insert the zone in the Structural Zones field, as shown below.

**Define Property Zones**

Create New Property Zone Method

Use Structural Zone(s)

Use Polygon Data Object

Name  
Property Zone 1

Description

Select one or more structural zones

Structural Zones

Property Zones

Property Zone 1

Save

Property Values


Group of parameters to define  
Conductivity (Kx, Ky, Kz)

|   | Parameter | Unit | Method   | Value  | Object | Mapping |
|---|-----------|------|----------|--------|--------|---------|
| ▶ | Kx        | m/s  | Constant | 0.0001 | →      |         |
|   | Ky        | m/s  | Constant | 0.0001 | →      |         |
|   | Kz        | m/s  | Constant | 1E-05  | →      |         |

- Select the **Group of parameters** that will be defined, e.g., Conductivity, Storage or Initial heads. The data input grid below will display the appropriate parameters based on which


parameter group is selected. For example, if conductivity is selected, the data input grid will show the parameters Kx, Ky, and Kz. The data input grid will already be populated with the default values specified in the Project Settings (File > Project Settings... ).

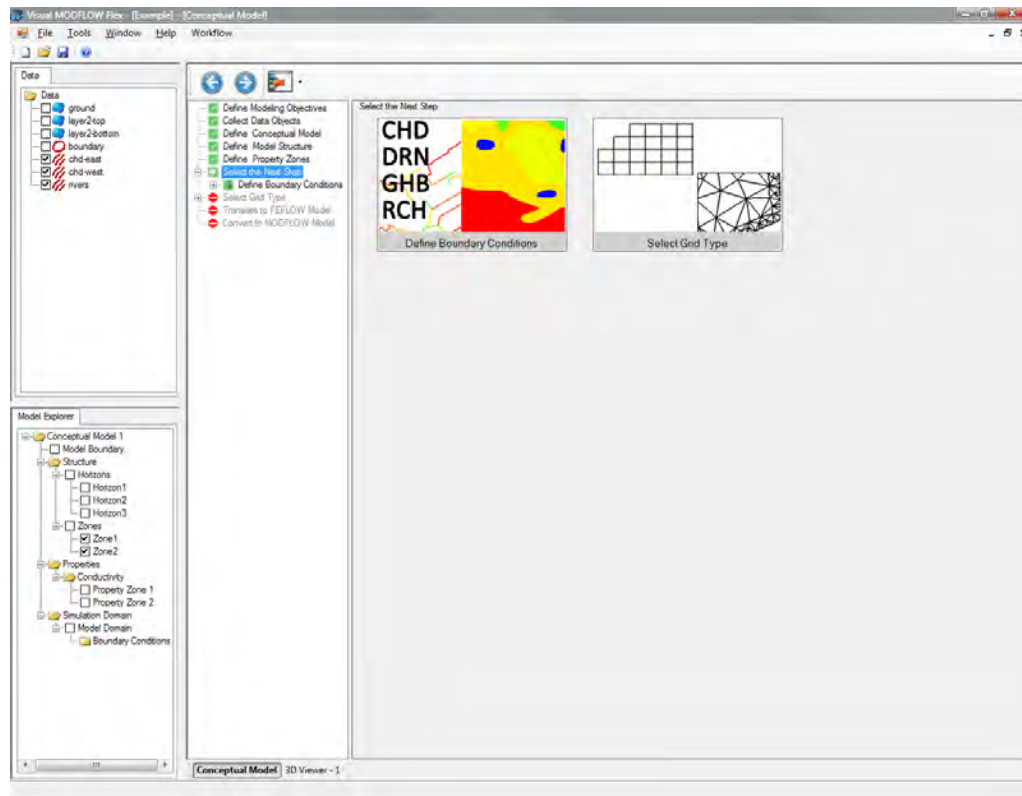
|   | Parameter | Unit | Method   | Value | Object | Mapping |
|---|-----------|------|----------|-------|--------|---------|
| ▶ | Kx        | m/s  | Constant | 4E-06 | →      | ▼       |
|   | Ky        | m/s  | Constant | 4E-06 | →      | ▼       |
|   | Kz        | m/s  | Constant | 4E-07 | →      | ▼       |

- Type the desired values for the property zone. (Kx = 4E-6, Ky = 4E-6, Kz = 4E-7)
- Click on the [Save] button located on the right side of the window.
- Repeat these steps for the other property zone:
- Click on the [Use Structural Zone] button
- Select Zone2 from the model tree
- Click on  button to insert the zone in the Structural Zones field, as shown below.
- Type the desired values for property zone2: (Kx = 7E-5, Ky = 7E-5, Kz = 7E-6)
- Click on the [Save] button located on the right side of the window.

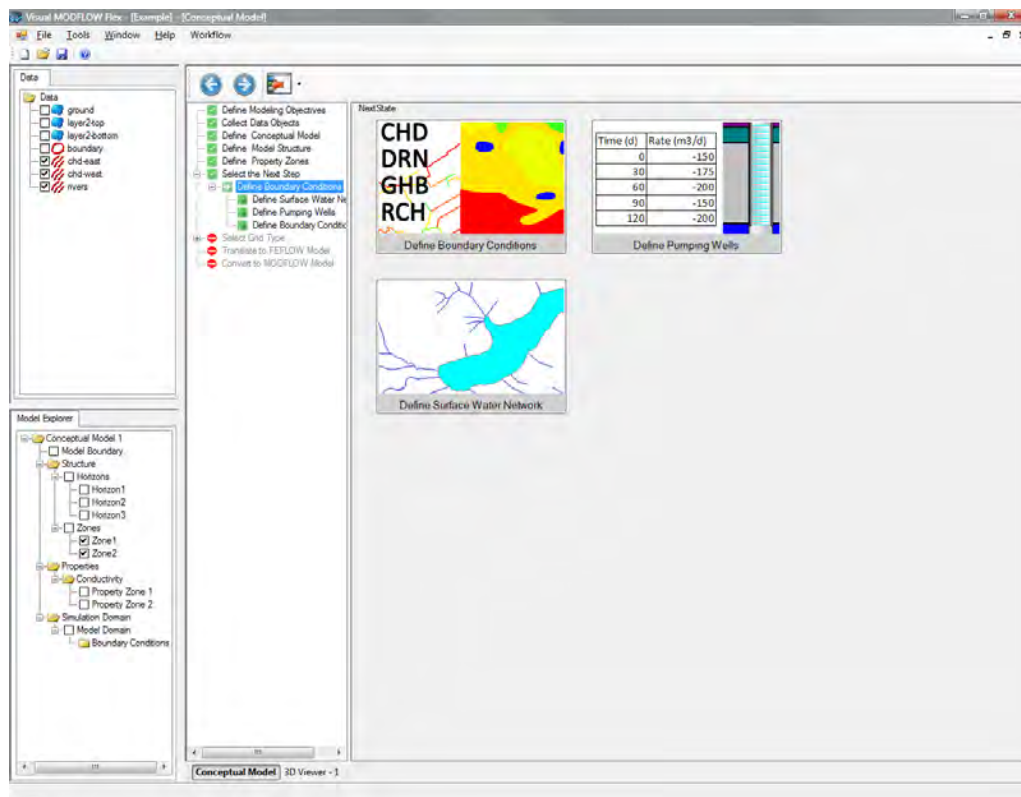
|   | Parameter | Unit | Method   | Value | Object | Mapping |
|---|-----------|------|----------|-------|--------|---------|
| ▶ | Kx        | m/s  | Constant | 7E-05 | →      | ▼       |
|   | Ky        | m/s  | Constant | 7E-05 | →      | ▼       |
|   | Kz        | m/s  | Constant | 7E-06 | →      | ▼       |

- Property zones can also be defined using polygon shapes; the values can also be defined from shapefile attributes or 2D Surface (distributed values). For more details, please see [Defining Property Zones](#) <sup>215</sup>

- Click  (Next Step) to proceed to the Selection screen.
- In this screen, you can choose to proceed to Defining Boundary Conditions or proceed to Defining a grid or mesh.
- Click the [Define Boundary Conditions] button to proceed.



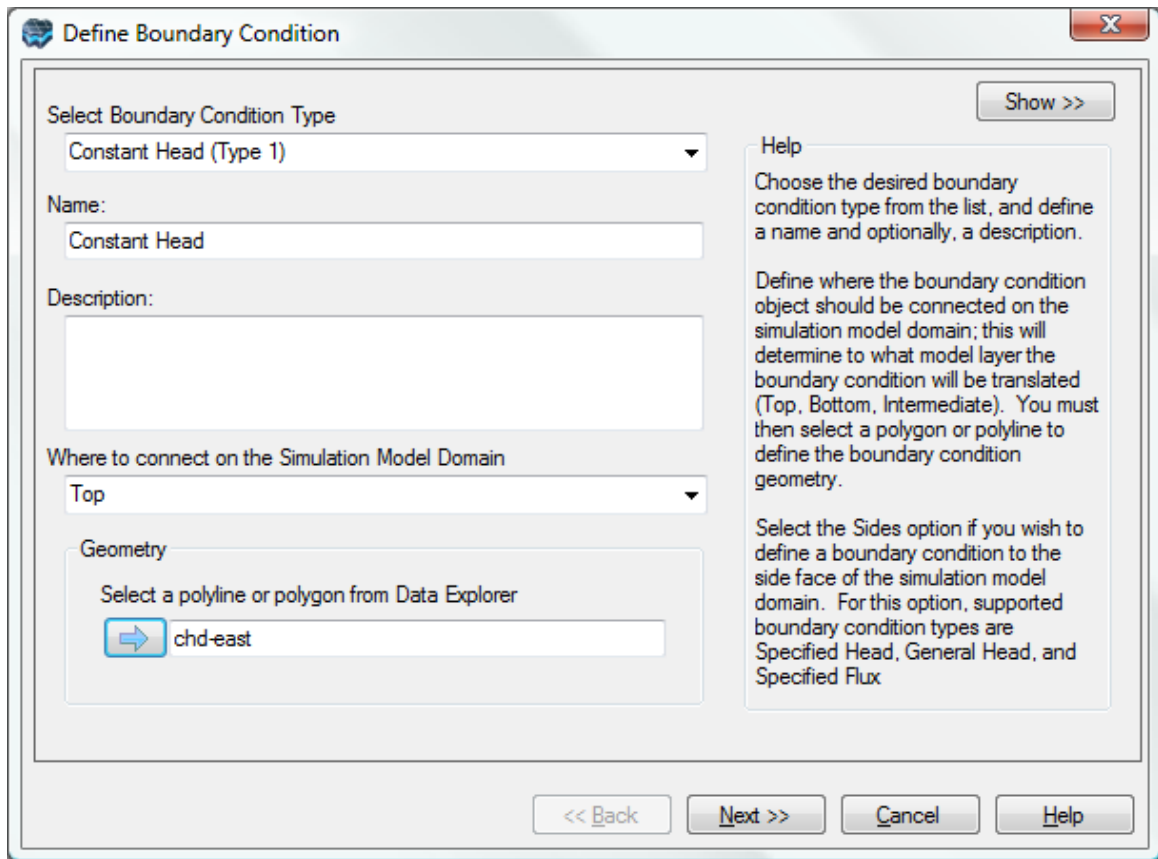
- In this window, you can choose the type of Boundary Conditions: Standard MODFLOW Boundary Conditions (CHD, DRN, RCH, etc.), Pumping Wells, or Surface Water Network (currently under construction).
- Click on the "Define Boundary Conditions" button:



- The Define Boundary Condition dialog box will appear on your screen as explained in the following section.

### Define Boundary Conditions

- At this step, you can define flow boundaries for the model.
- From the **Select Boundary Condition Type** combo box, select the desired boundary condition type.
- Constant Head
- Type name: Constant Head East



The image shows a software dialog box titled "Define Boundary Condition". It contains several input fields and a help section. The "Select Boundary Condition Type" dropdown is set to "Constant Head (Type 1)". The "Name" field contains "Constant Head". The "Description" field is empty. The "Where to connect on the Simulation Model Domain" dropdown is set to "Top". The "Geometry" section has a button with a blue arrow pointing right, and a text field containing "chd-east". The help section on the right provides instructions on how to choose a boundary condition type, define its name and description, and select the geometry to connect to. At the bottom, there are buttons for "<< Back", "Next >>", "Cancel", and "Help".

**Define Boundary Condition**

Select Boundary Condition Type  
Constant Head (Type 1)

Name:  
Constant Head


Description:

Where to connect on the Simulation Model Domain  
Top

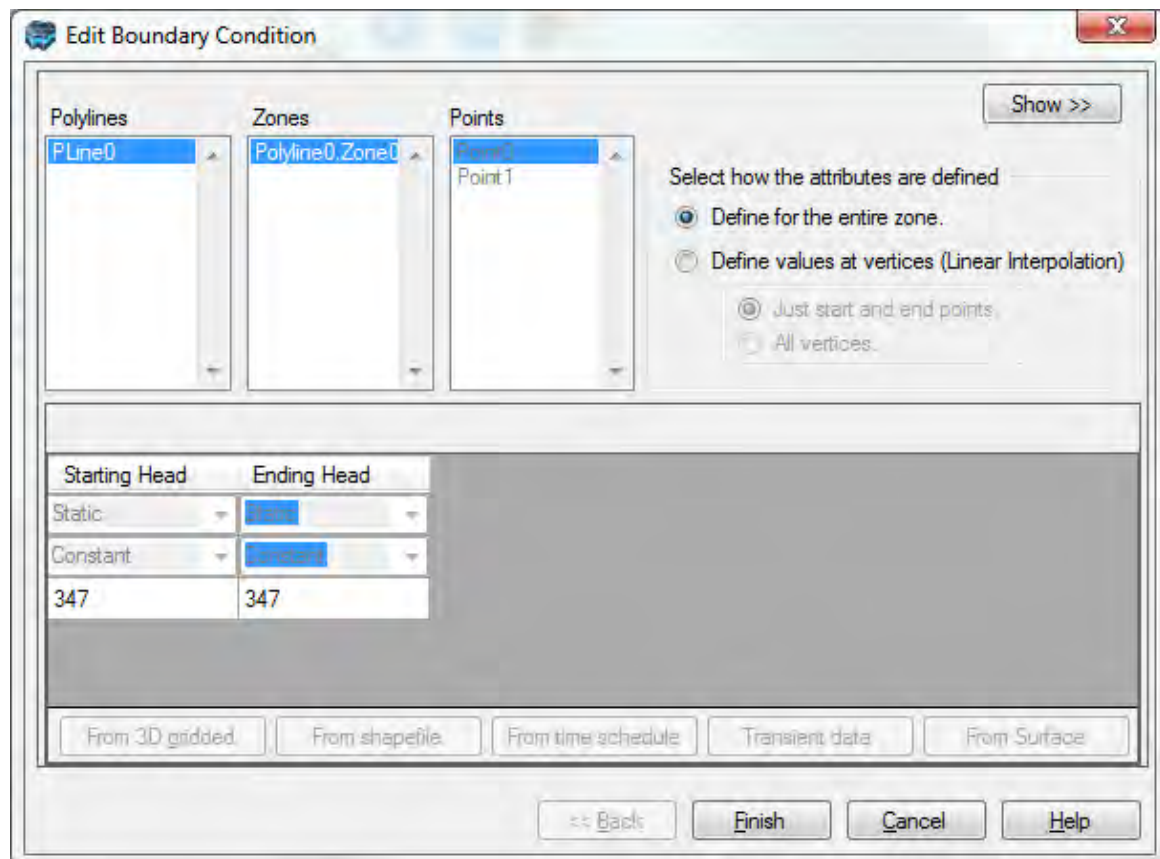
Geometry  
Select a polyline or polygon from Data Explorer  
chd-east

Help  
Choose the desired boundary condition type from the list, and define a name and optionally, a description.  
  
Define where the boundary condition object should be connected on the simulation model domain; this will determine to what model layer the boundary condition will be translated (Top, Bottom, Intermediate). You must then select a polygon or polyline to define the boundary condition geometry.  
  
Select the Sides option if you wish to define a boundary condition to the side face of the simulation model domain. For this option, supported boundary condition types are Specified Head, General Head, and Specified Flux

<< Back   Next >>   Cancel   Help

- From the Data Explorer, select the **chd-east** polyline that represents this constant head.
- Click the  button in the **Define Boundary Condition** dialog, to add this polyline to the input.
- Click the [Next] button.






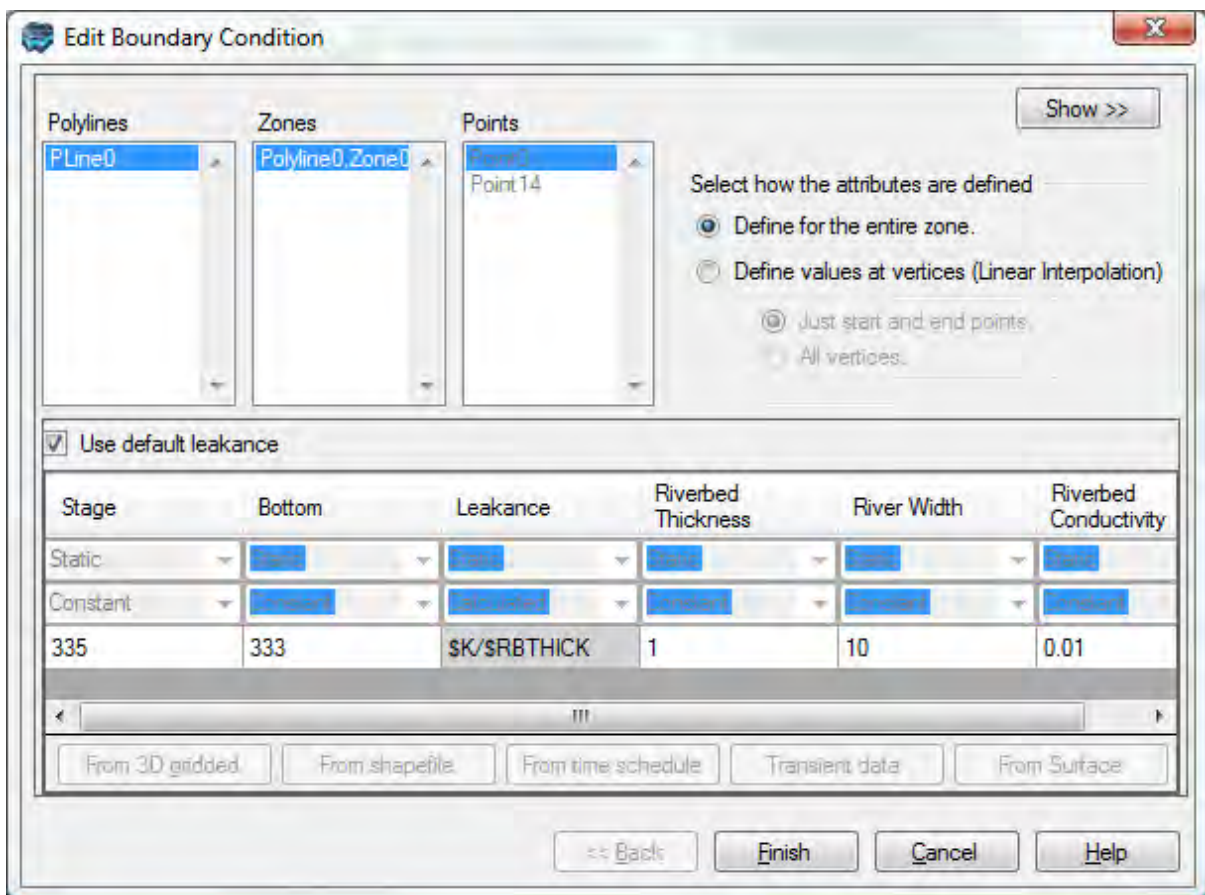
- The next dialog allows us to define the constant head value. VMOD Flex provides various options for defining boundary condition attributes. Attributes can be assigned from those stored in Surface, Time Schedule, Shapefile and 3D Gridded data objects. You can also set attributes as Static (no change over time) or Transient (changes over time).
  - For this tutorial, you will assign a static constant head value.
  - In the empty field located below the Starting Head field
  - type: 347 for the Starting Head and Ending Head
  - [Finish] button

Repeat these steps to define the other constant head boundary condition:

- Click on "Define Boundary Conditions" in the tree
- Select the "Define Boundary Condition" button.
- Choose Constant Head, select the chd-west polyline, and define a value of 325 for both the Starting Head and Ending Head
- Click Finish

The following display will appear. Before you proceed, you will define one more boundary condition, a River.

- Click on "Define Boundary Conditions" in the tree, and select the "Define Boundary Condition button".
- Choose River for the boundary condition type
- From the Data Explorer, select the "river" polyline
- Click the  button in the Define Boundary Condition dialog, to add this polyline to the input.
- A warning may appear about clipping the polyline; click OK to continue
- Click the [Next] button.
- Define the following attributes for the river: Stage: 335, Bottom: 333, Riverbed Thickness: 1, Width: 10, Riverbed conductivity: 0.01 (m/s)

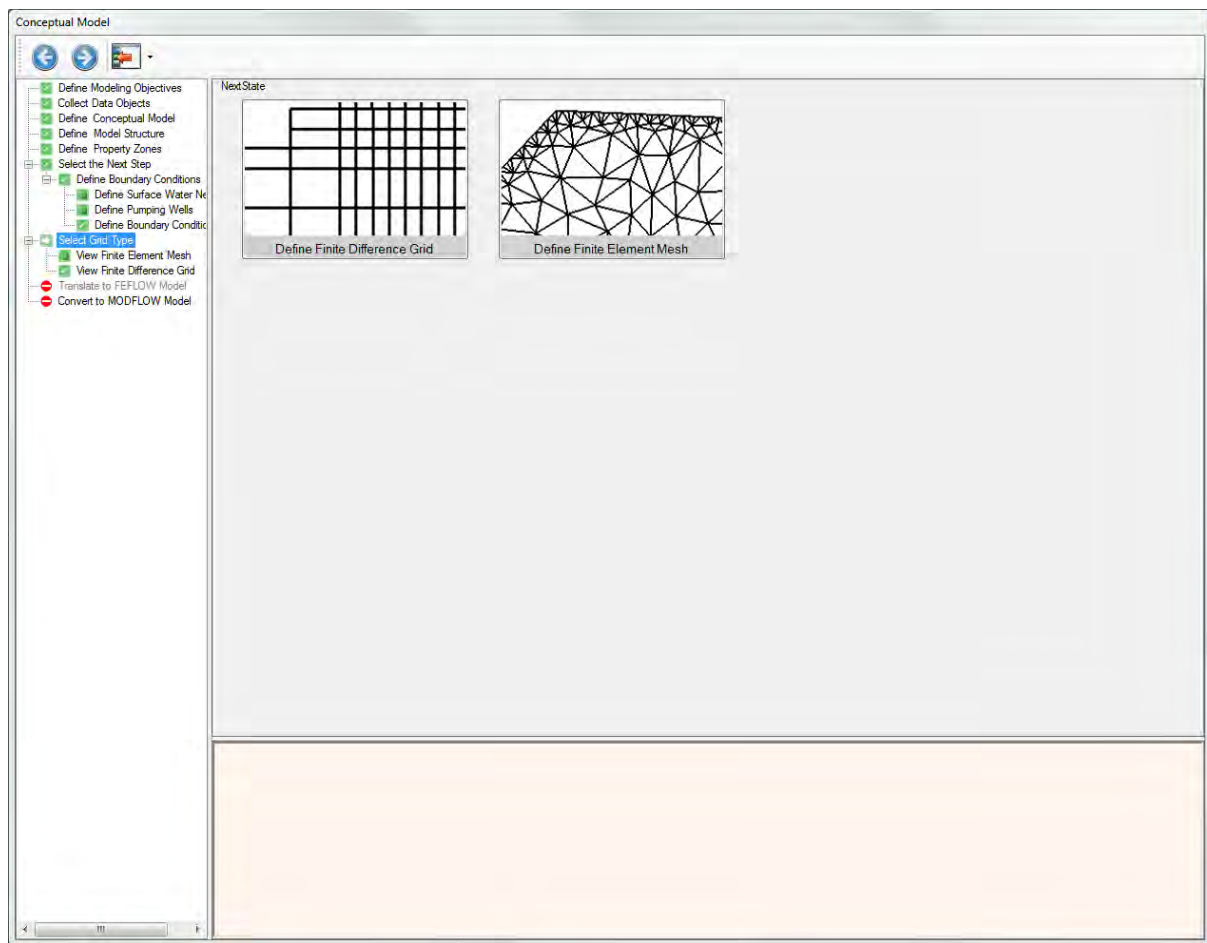


The dialog box titled "Edit Boundary Condition" contains three lists: Polylines (PLine0), Zones (Polyline0.Zone0), and Points (Point0, Point14). To the right, under "Select how the attributes are defined", the radio button "Define for the entire zone." is selected. Below this, the checkbox "Use default leakance" is checked. A table defines attributes for the boundary condition.

| Stage    | Bottom   | Leakance      | Riverbed Thickness | River Width | Riverbed Conductivity |
|----------|----------|---------------|--------------------|-------------|-----------------------|
| Static   | Static   | Static        | Static             | Static      | Static                |
| Constant | Constant | Constant      | Constant           | Constant    | Constant              |
| 335      | 333      | \$K/\$RBTHICK | 1                  | 10          | 0.01                  |

At the bottom of the table are five buttons: "From 3D gridded", "From shapefile", "From time schedule", "Transient data", and "From Surface". At the very bottom of the dialog are four buttons: "Back", "Finish", "Cancel", and "Help".

- Click Finish. The River conceptual boundary condition will be added to the model tree. The following display will appear.



Next you can define a Finite Difference Grid or a Finite Element Mesh

- Click the [Define Finite Difference Grid] button and define the inputs as explained in the following section.

### Define Grid

Define Numerical Grid

Name  
NumericalGrid1

Define Horizontal Grid

Rotation  
0

Grid Size

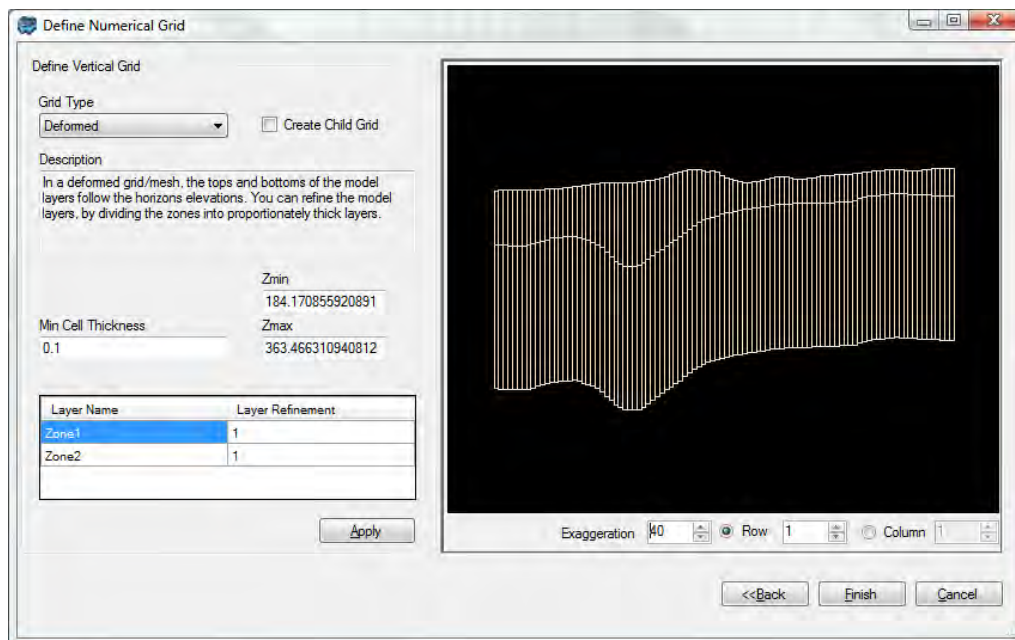
|         |             |
|---------|-------------|
| Rows    | Cell Height |
| 100     | 130.000     |
| Columns | Cell Width  |
| 100     | 130.000     |

Grid Extents

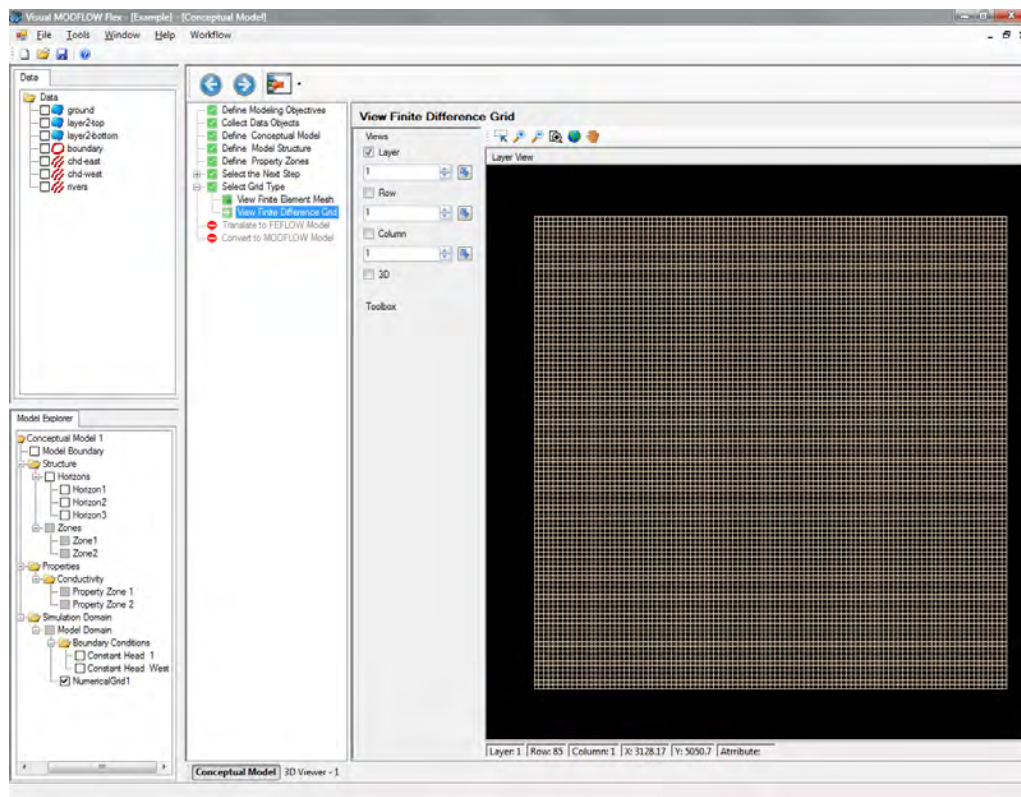
|          |           |           |
|----------|-----------|-----------|
| Xmin     | Xmax      | Width     |
| 3000.000 | 16000.000 | 13000.000 |
| Ymin     | Ymax      | Height    |
| 3000.000 | 16000.000 | 13000.000 |

<<Back Next >> Cancel

- Enter a unique Name for the numerical grid. This name will appear in the Conceptual Model tree once the grid is created.
- Enter the grid size, and optionally, the grid rotation. The grid can be rotated counter-clockwise about the grid origin by entering a value between 0 and 360 in the Rotation text field.
- The Xmin and Ymin values refer to the X-Y coordinates of the bottom-left corner of the numerical grid. The Xmax and Ymax values refer to the X-Y coordinates of the top-right corner of the numerical grid.
- The Columns and Rows fields allow you to define the Grid Size.
- Type: 100 for both the # rows and columns
- Click the [Next] button to proceed to define the vertical discretization.



- In the "Define Vertical Grid" screen, specify the type of vertical discretization; for this exercise, the default Deformed grid will be used. More details on the grid types can be found in the [Defining Grids/Meshes](#) <sup>251</sup> section.
- Leave the defaults as is; click the [Finish] button. The Grid will then appear as shown in the following screen.

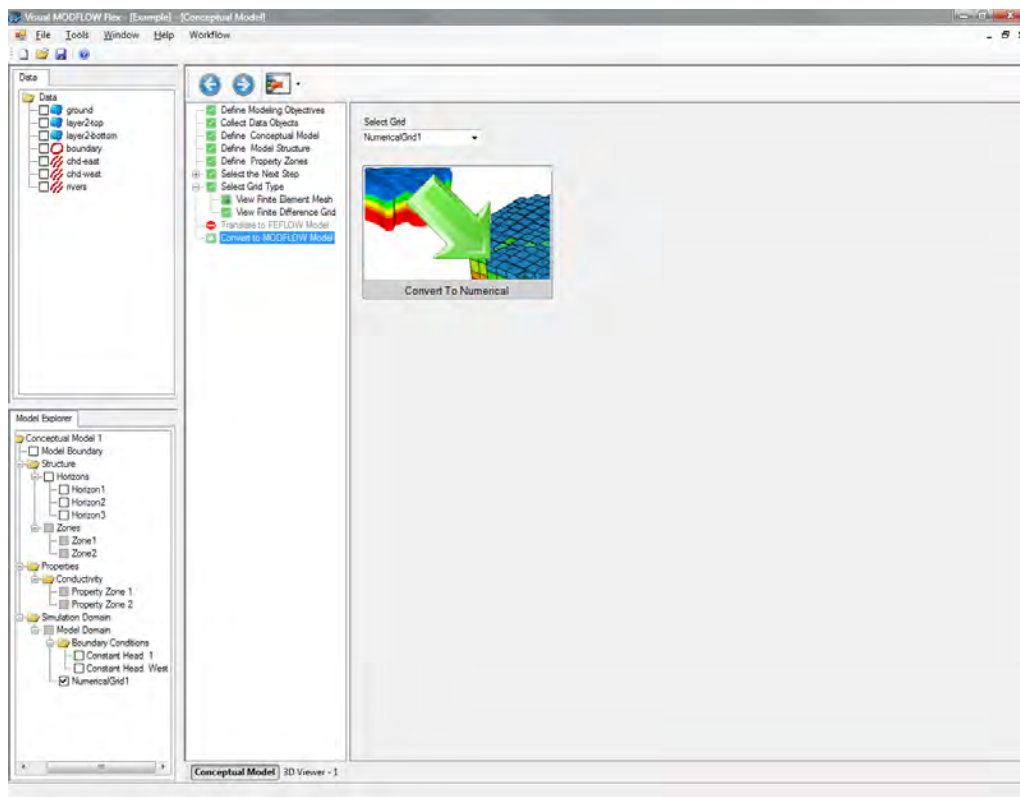


- Click  (Next Step) to proceed.

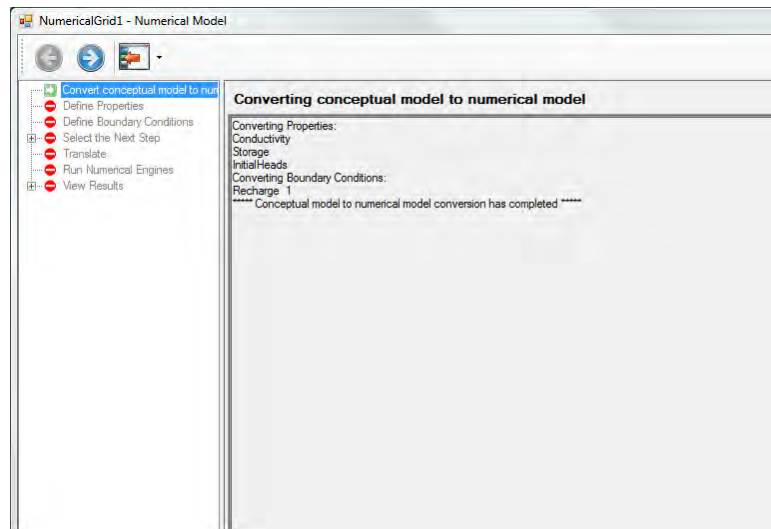
### Convert to Numerical Model

Now you are ready to populate the numerical grid/mesh with the conceptual elements.

- Click on the "Convert to Numerical Model" button to proceed



- After clicking on the conversion button, a new window will appear which includes the steps for the numerical model. In the first window, you will see the progress of the conceptual to numerical conversion. This conversion could take several minutes, depending on the size and type of grid you used, and the complexity of the conceptual model inputs.

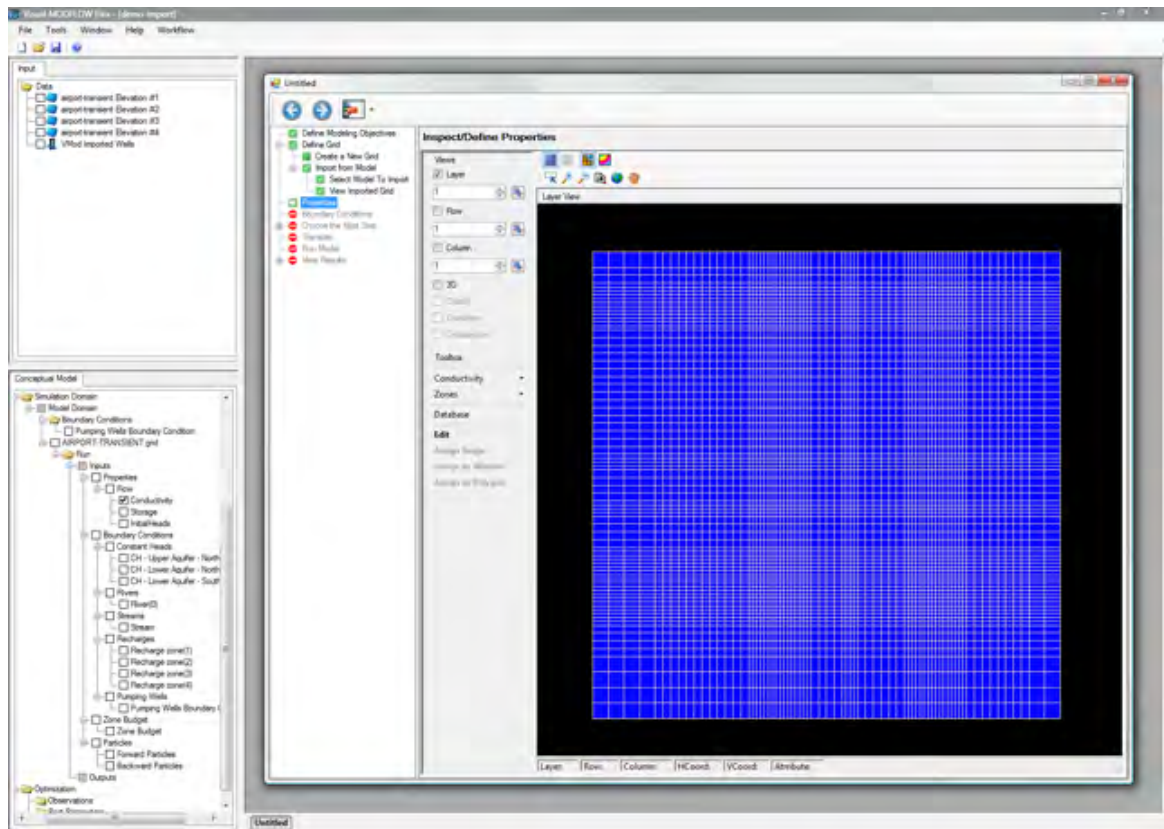





- When the conversion is complete, click  (Next Step) to proceed to the Properties step.

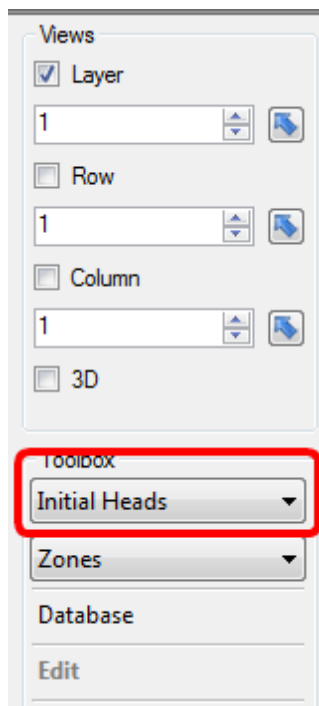
### View/Edit Properties

- At this step, you can view/edit the flow properties for the model.








- Under Views, select the various views you want to see in the Flex viewer; VMod Flex allows you to simultaneously show a layer, row, column and 3D Views. Place a checkbox beside the desired view and it will appear on screen.
- Adjust a specific layer, row, or column using the up/down arrows. Alternatively, click on the  button then click on any specific row, column, or layer in any of the 2D views, and the selected row, column, or layer will be set automatically.
- Now you will define a default initial heads value.
- Choose "Initial Heads" from the combo box under the Toolbox as shown below





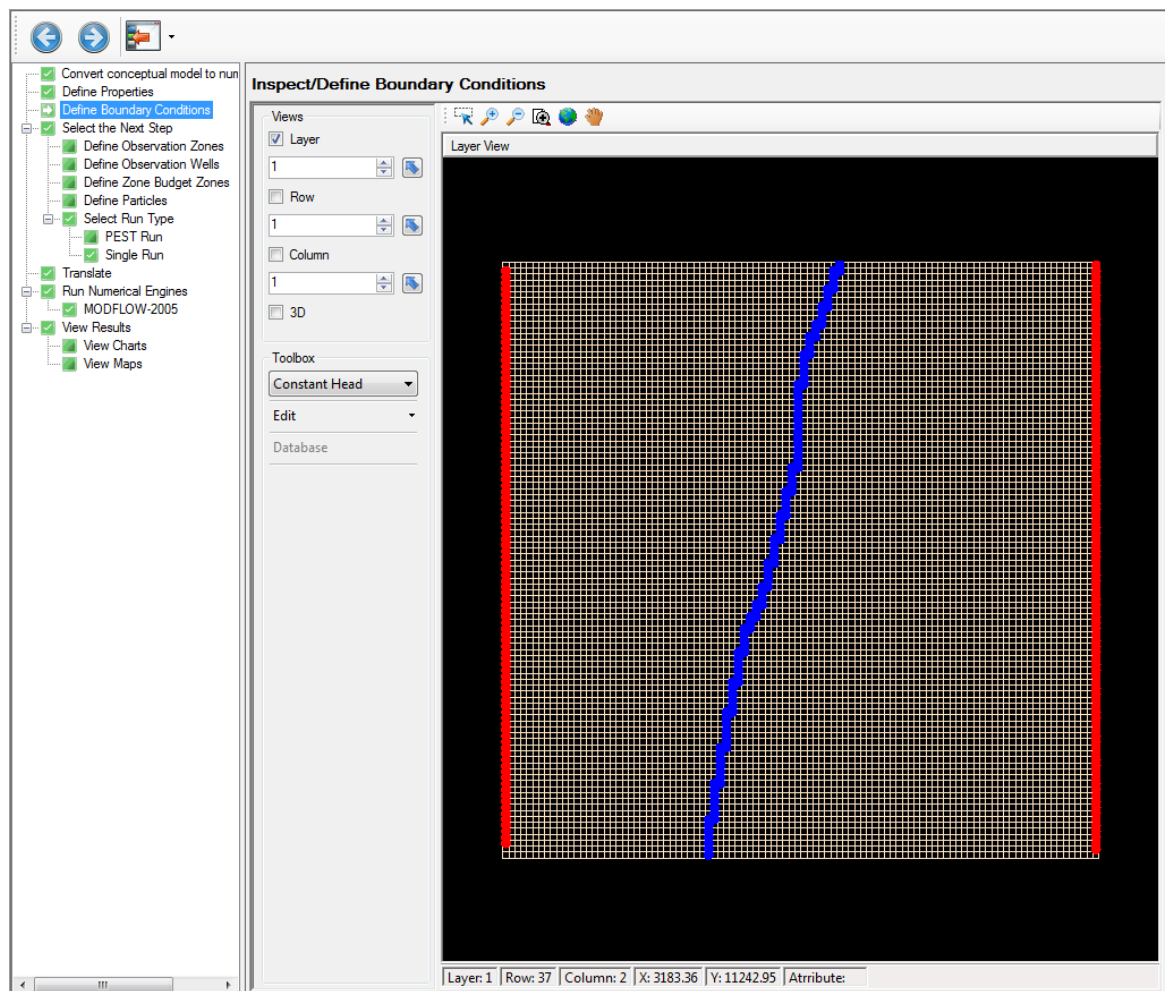
- Click [Database] button located below the Initial Heads combo box.
- Type: 350 for each of the zone values; this will apply an initial head value of 350 for the entire model domain.
- Click OK when you are finished.
- Use the same tools as described in the previous step to manipulate the views.
- The display tools will allow you to switch from discrete cells rendering to color shading/contours.


-  Render as cells
-  Show color shading
-  Show/hide contour lines
-  Show/hide grid lines

- In the Toolbox, you can select a different parameter group (for example Initial Heads), and see the corresponding zonation in the Flex Viewers.
- Click  (Next Step) to proceed to the Boundary Conditions step.

### View/Edit Boundary Conditions

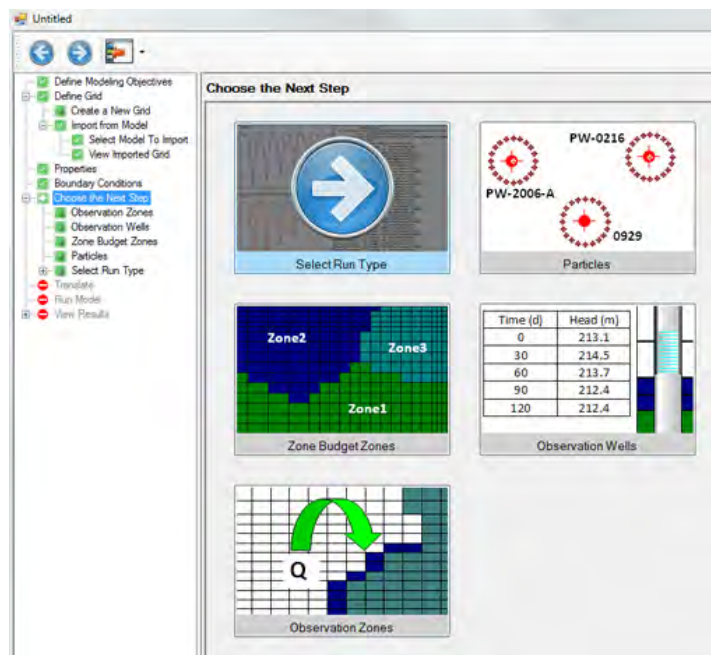
- At this step, you can view/edit the flow boundaries for the model.




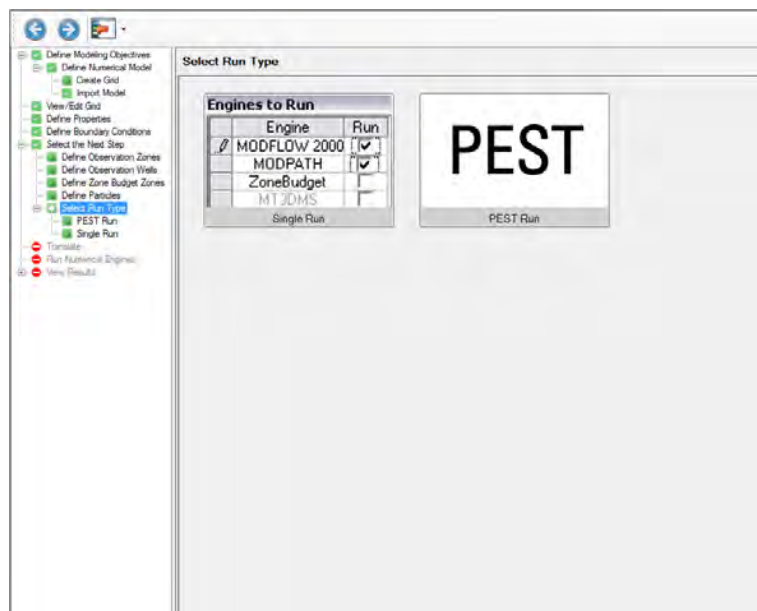
- From the toolbox, select the Desired Boundary condition group (Constant Head, Rivers, etc..)
- Then select Edit (Single) or Edit (Group).
- Click on a cell that belongs to this group; a dialog will appear where you can see the parameters for a single cell or group of cells.
- Click  (Next Step) to proceed. You will arrive at the Flow Boundaries step


#### [Proceed to Run or Define Optional Model Elements](#)

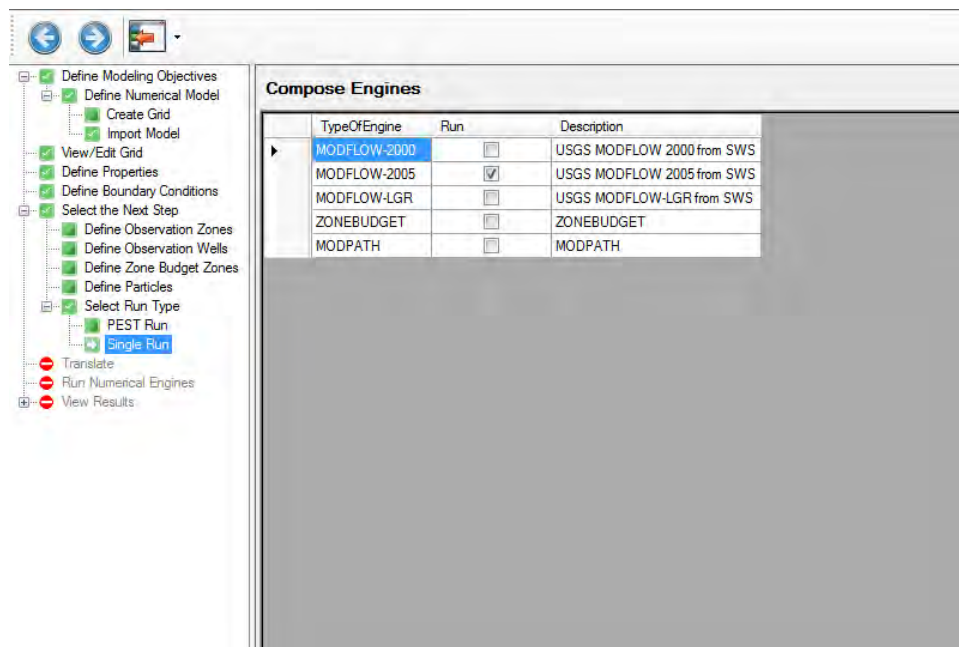
- You will arrive at a choice screen; here you can proceed to some of the “non-essential” inputs for the model, such as Zone Budget Zones, Particle Tracking, or Observation Wells. Or, you can proceed to Running the simulation




- Click the “Select Run Type” button to proceed (Mouse over this and you will see the blue "Next" arrow appear on top; just left click once to select this option. (Alternatively, the  (Next step) button will take you to this step, as it is pre-defined as the default step.



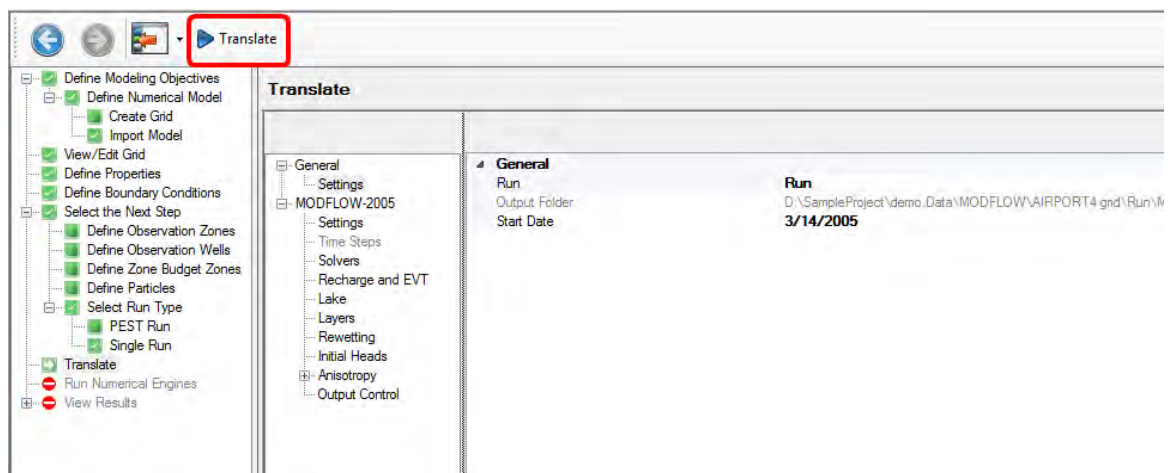
- Click the “Compose Engines” button to proceed (Alternatively, the  (Next step) button will take you to this step).



- You will arrive at the “Select Engines” step. Here you can choose what engines you want (what version of MODFLOW: 2000, 2005, etc..), and if you want to include MODPATH and ZoneBudget in the run
- MODFLOW-2005 should be selected by default.
- Click  (Next Step) to proceed.

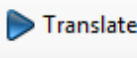
## Translate Packages

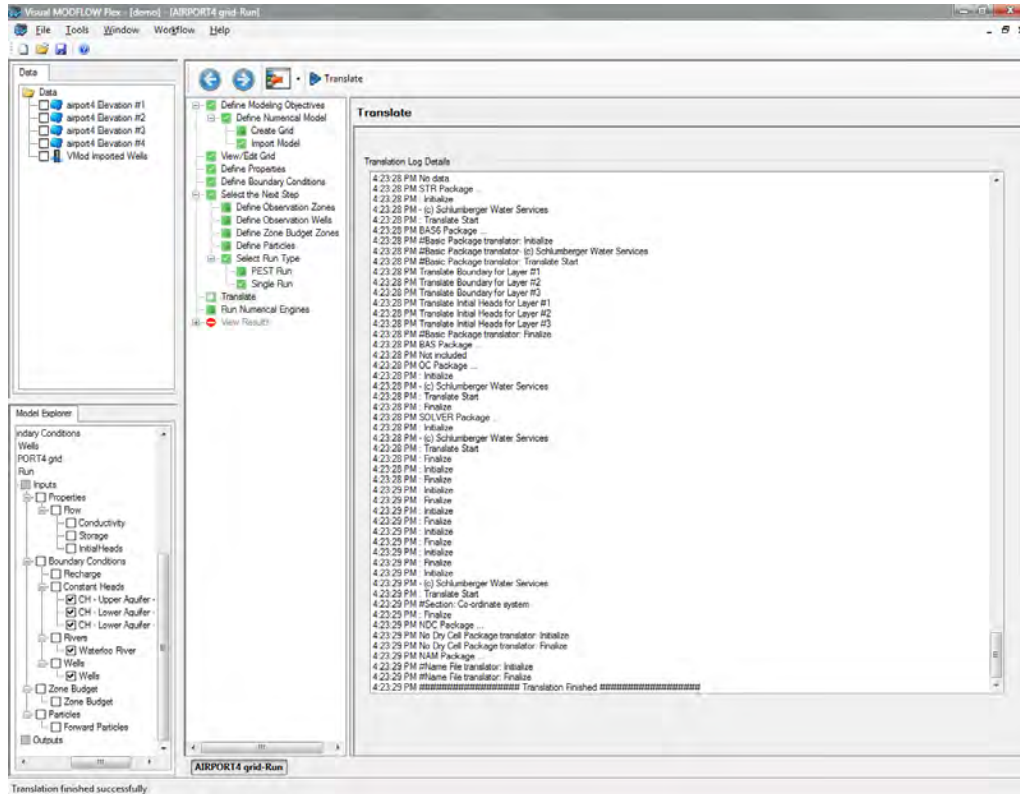
- You will arrive at the “Translation Step”.




At this step, you choose if the model is steady-state or transient, choose the solver you

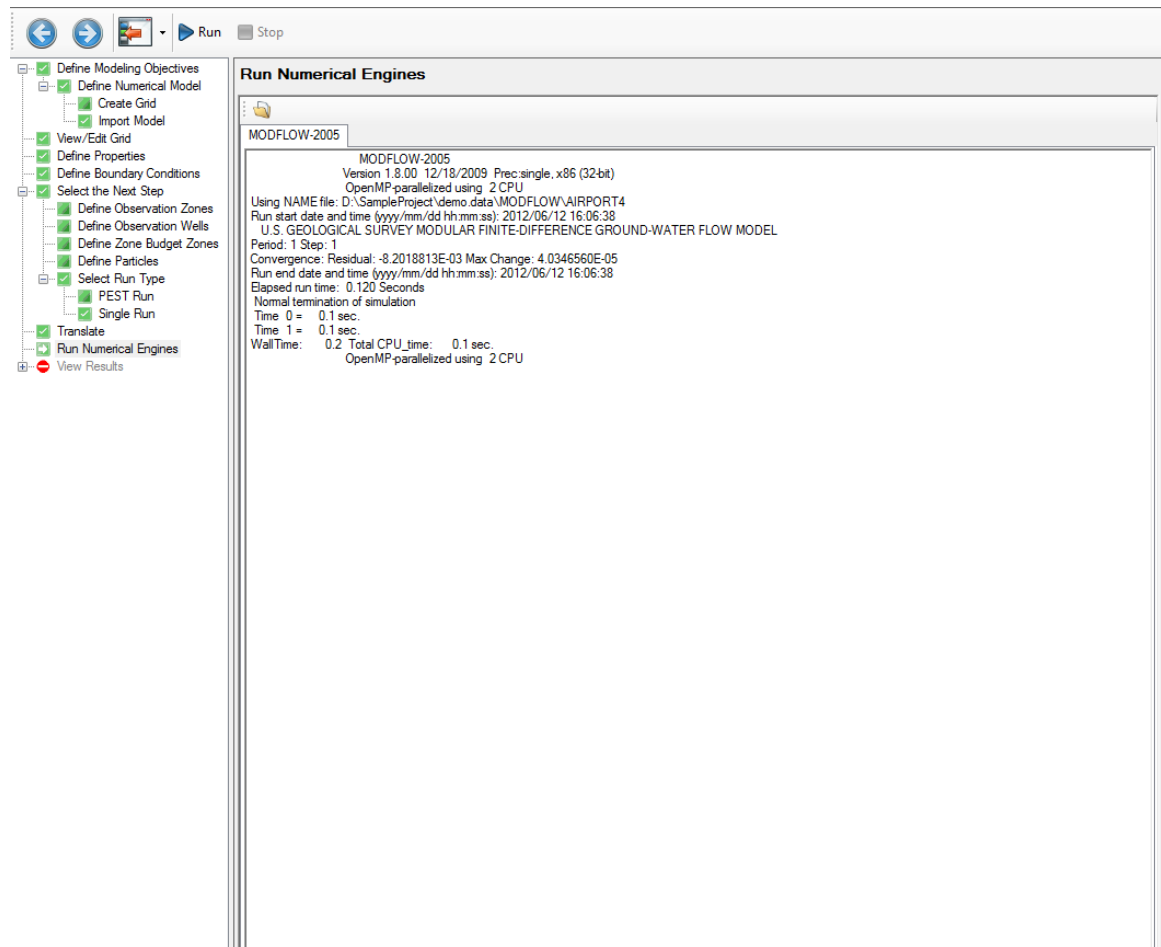
want to use, and define any other MODFLOW package/run settings, such as cell-rewetting, etc. Note in the General Settings, there is a default location where the MODFLOW and other files will be generated; this can be modified if you wish.



- Click the  button (circled above) to proceed; this will read the input from the numerical model and “translate” this into the various input files needed by MODFLOW and the other engines. The files will be created in the directory defined in the previous step.



- Click the  (Next step) button to proceed. You will arrive at the “Run Engines Step”.

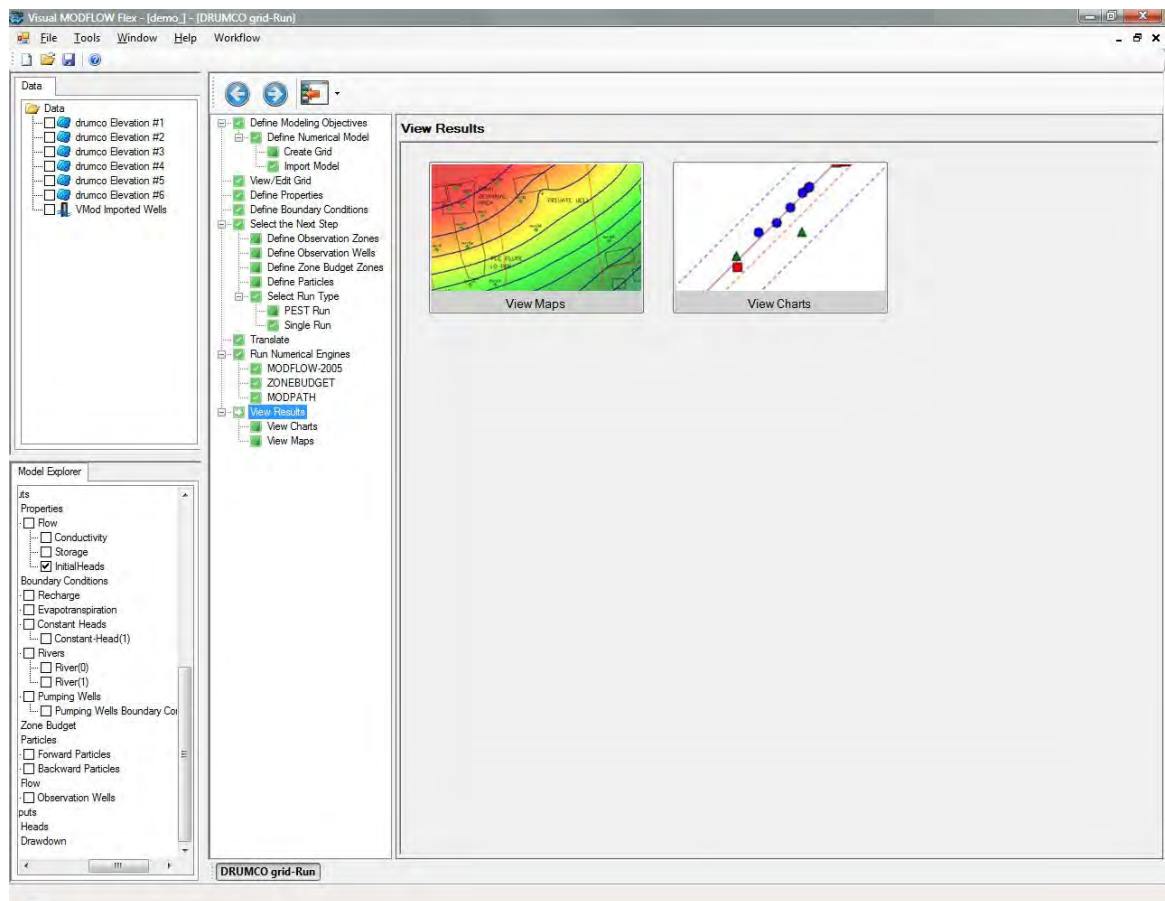
## Run Engines



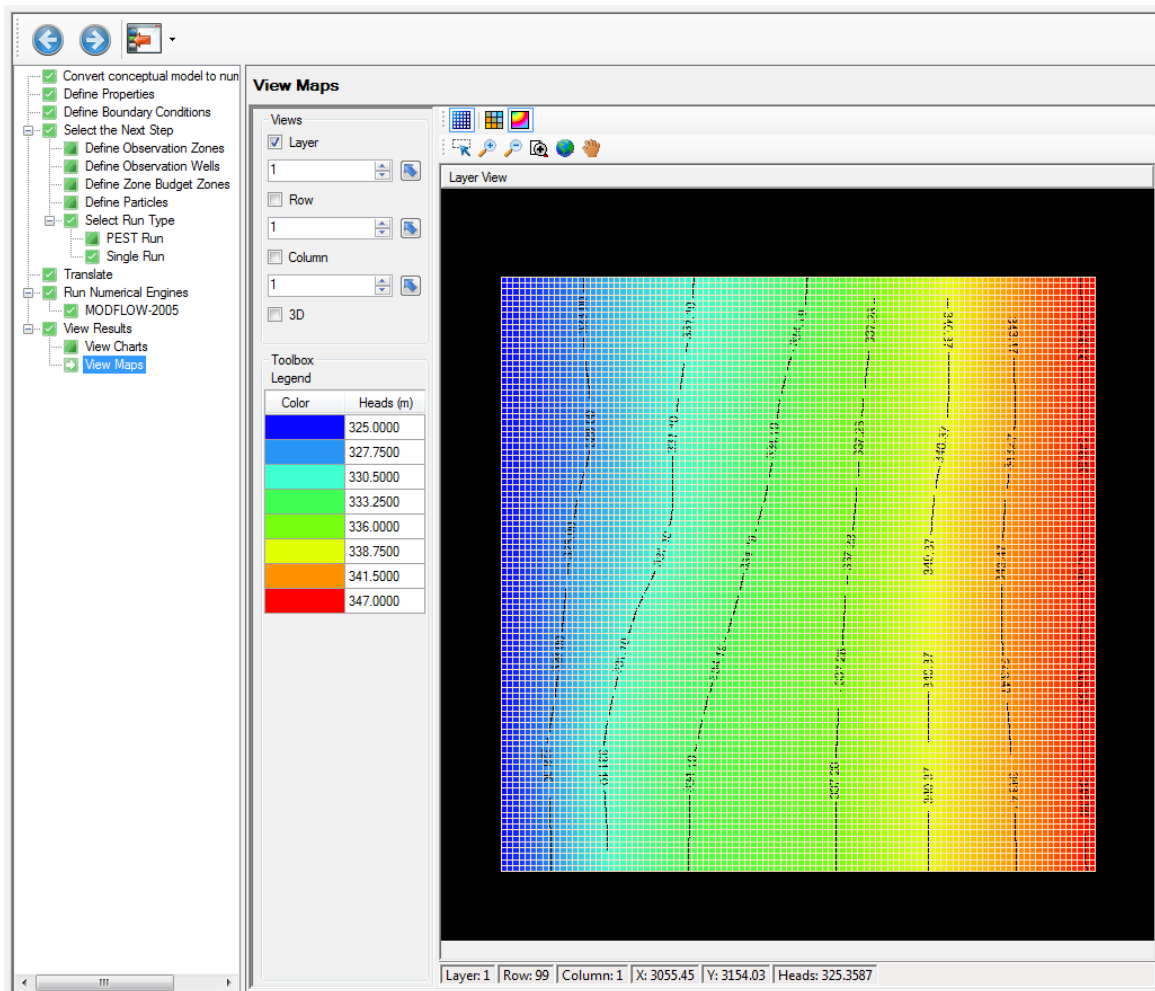
- Click the  button on the main workflow toolbar to start running the engines. You will see the Engine progress in the scrolling window.
- Note that after a successful run, the Heads and Pathlines items will be added the tree in the model explorer.
- Once finished, Click the  (Next step) button to proceed.

### View Results

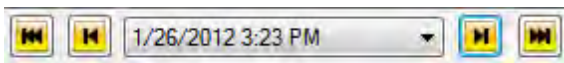
- You can then choose to view results in the form of Maps (Contours and Color shading) or Charts



- Click the "View Maps" button.



- You will then see color shading of the calculated heads, in layer view.
- You can display heads along a row, and along a column, and in 3D, using the same tools as you used earlier. (refer to [View/Edit Properties](#) <sup>[40]</sup> section)
- If your model is transient, use the time controls above the Flex Viewer to change the output time; as you do this, all active viewers (layer, row, column, 3D), will refresh to show the heads for the new output time.



- The next section will discuss how you can generate a new grid with a different size and resolution, and generate a numerical model using this grid.

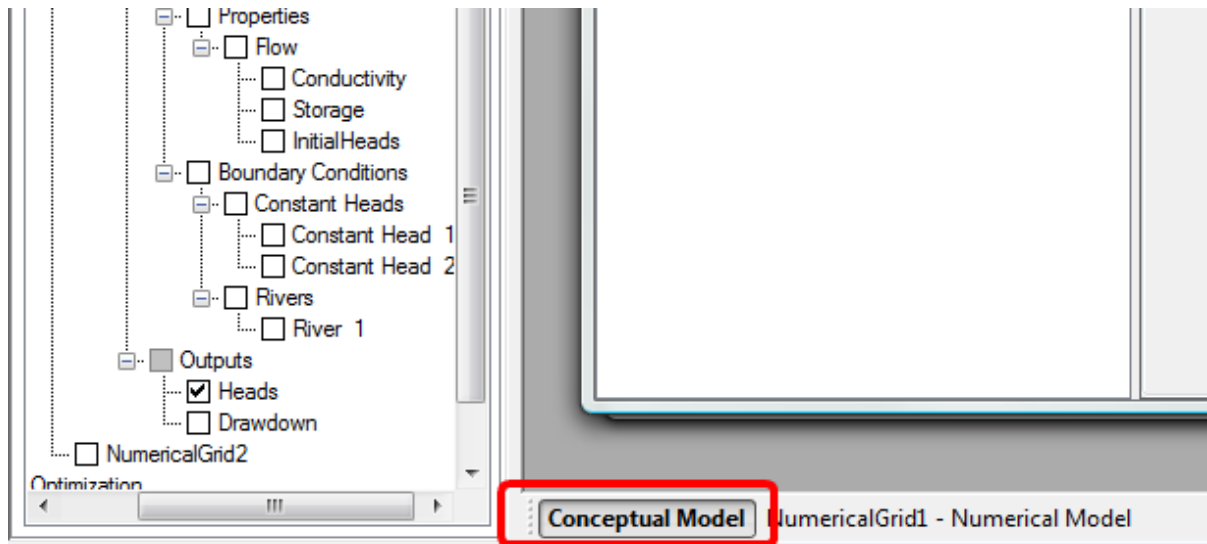
### Evaluating Different Grids


Often the initial grid size you defined is not adequate to provide the solution and stability you



require from your model. In this section, we will explain how you can generate multiple grids from the conceptual model and run the corresponding numerical models.

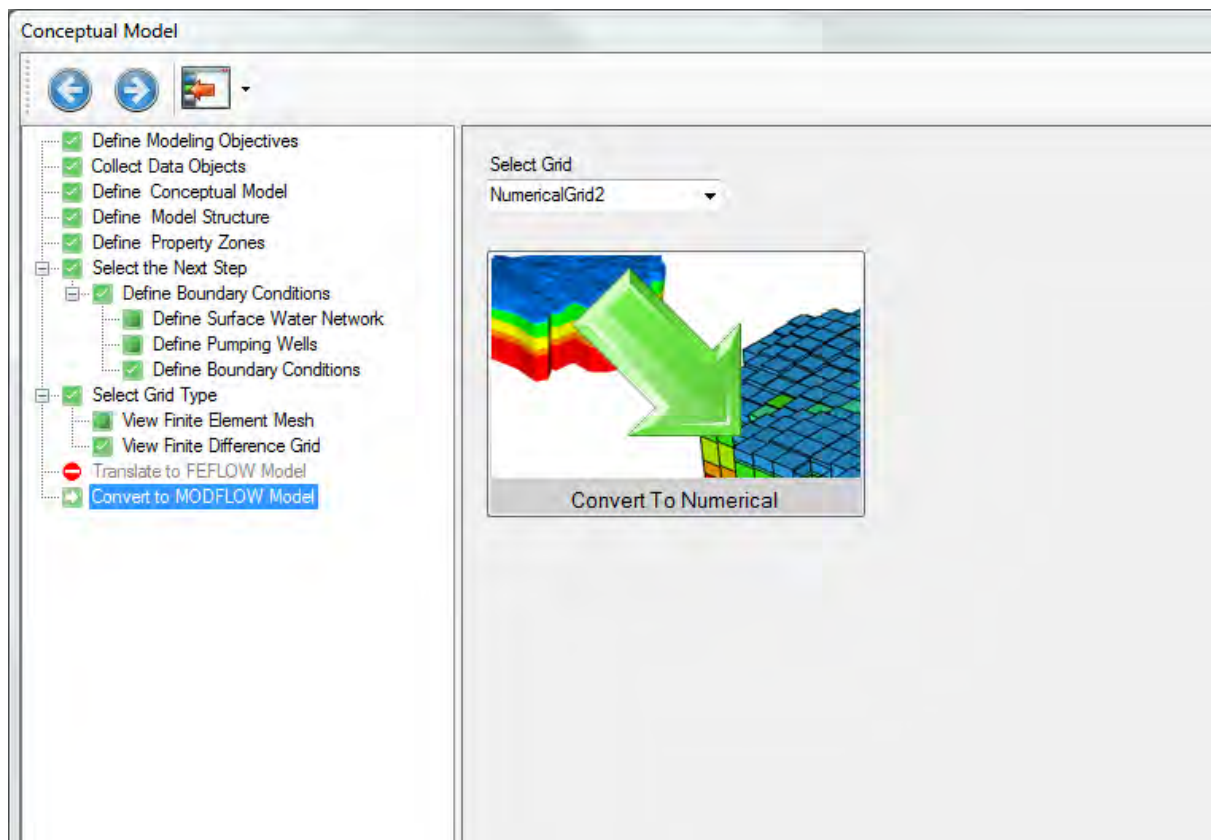
- Below the windows, near the bottom of your display, you will see a list of active windows.



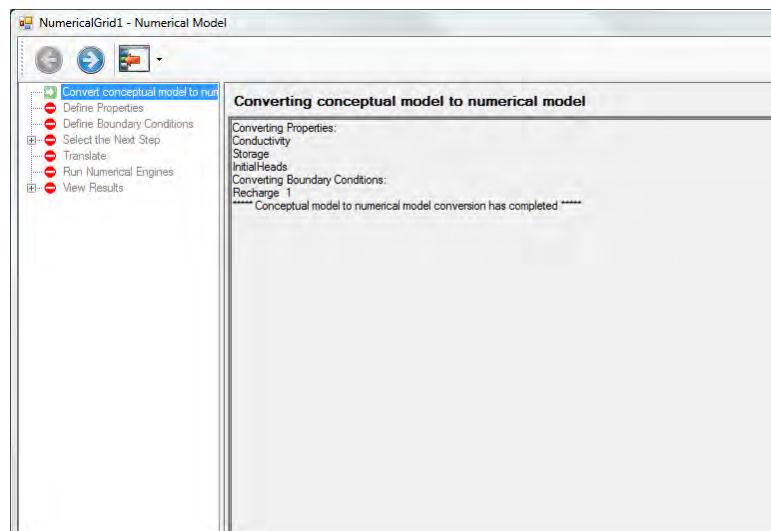
- Click on the Conceptual Model button to make this the active window, and it should now appear on your display.
- Click on "Select Grid Type" from the workflow tree.
- Click [Define Finite Difference Grid] button and the Define Grid window will appear.
- Define a new grid with the desired grid size and rotation.
- Click [Next]
- Specify the desired vertical discretization; you may wish to use a different vertical grid type, or refine any of the vertical layers.
- Click [Finish] when you are done.
- The new grid should now appear, and you will also see the grid appear as a new node in the model tree.
- Click the  (Next step) button to proceed.

Now you are ready to populate the numerical grid/mesh with the conceptual elements. The "Convert to Numerical Model" display should appear similar as below. Now, in the "Select Grid" combo box, you will see there are 2 grids; by default, the grid you created should be selected.

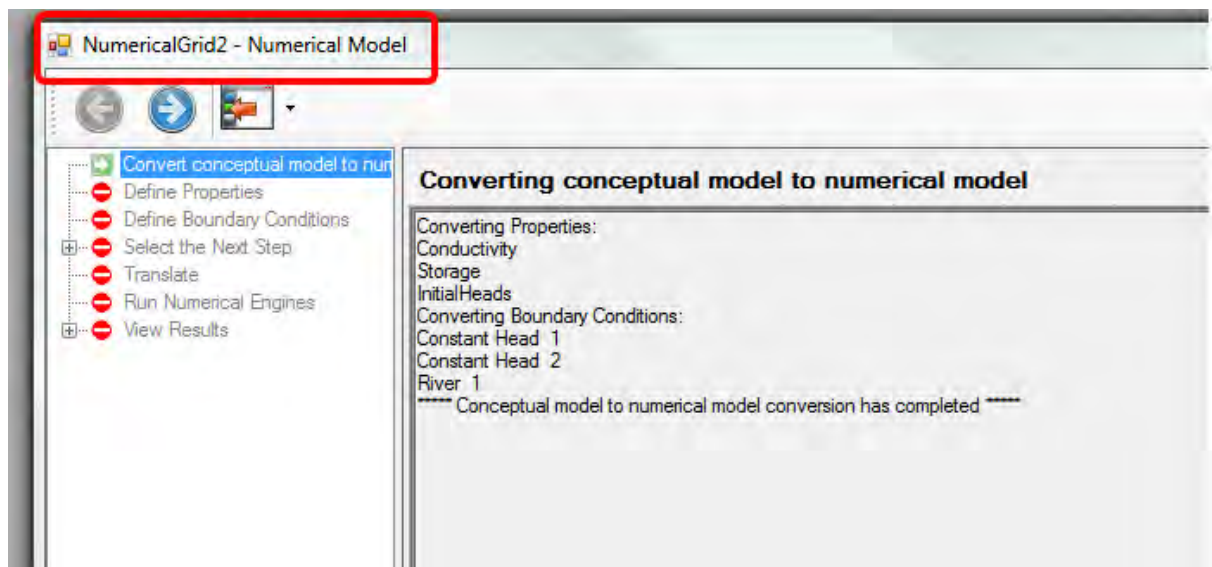
- Click on the "Convert to Numerical Model" button to proceed.



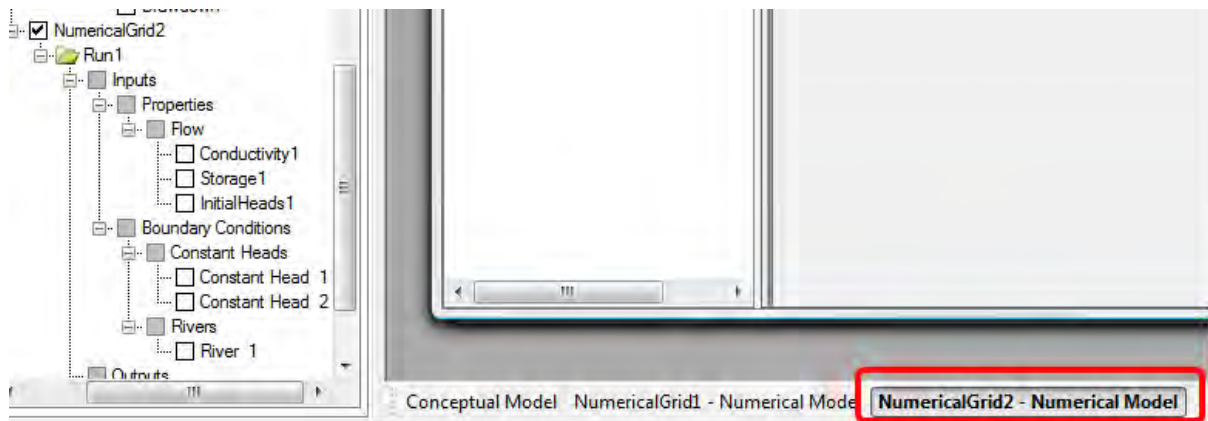
- After clicking on the conversion button, a new workflow window will appear which includes the steps for the numerical model for this new grid. In the first window, you will see the progress of the conceptual to numerical conversion. This conversion could take several minutes, depending on the size and type of grid you used, and the complexity of the conceptual model inputs.




- Note that the new window is titled with the name of the new grid you provided.



- This new window will appear in the active list at the bottom of the main parent window.



- In addition, this new model run will appear in the model tree. The model run has a grid and corresponding inputs; this can also be seen in the figure above.
- When the conversion is complete, click  (Next Step) to proceed to the Properties step.
- Now, as explained previously, you can review the properties and boundary conditions, and translate and run this model.
- Once the heads are generated, you can compare this to the results from previous grids.

### 3.2 Importing VMOD/MODFLOW Models

The following example is a quick walk through of the basics of importing an existing Visual MODFLOW or MODFLOW data set.

#### Objectives

- Learn how to create a project and import an existing numerical model
- Become familiar with navigating the GUI and steps for numerical modeling
- Learn how to view and edit properties and boundary conditions, in a variety of views
- Translate the model inputs into MODFLOW packages and run the MODFLOW engines
- Understand the results by interpreting heads and drawdown in several views
- Check the quality of the model by comparing observed heads to calculated heads

#### Required Files

- This tutorial is designed to allow you to select your own Visual MODFLOW or MODFLOW project, and follow through the steps. If you wish to use the model that is shown in the following example, it can be downloaded from our website below:

<http://trials.swstechnology.com/software/VMODFlex/2012/Tutorials/vmod-model-import.zip>